It's 10 o'clock. Do you know where your files are?

Our P: drive is like a big closet. Since we don't have unlimited space, we need to clean it out occasionally. So, files are removed from P: for one of two reasons: 1. files are inactive for a period of about one year, or 2. project phases are complete, ready for archiving in TIC.

We must also keep our closet organized. **Please remember to store files under the established folder structure:**

P:\[park]\[package]\[phase]

Archive Database

If you cannot find files on P:, and suspect they've been archived, look in our archive database. Find it at P:\Cadd_adm\Archive\Archive.mdb. Using Windows Explorer, double click on that filename, which will launch Microsoft Access. Or Start Microsoft Access and open the database.

If you need files restored, call me with the CD ID# (find the CD ID# in the archive databse), and I'll be happy to restore files or provide you with a duplicate of the entire CD.

Do you have a project ready for archiving?

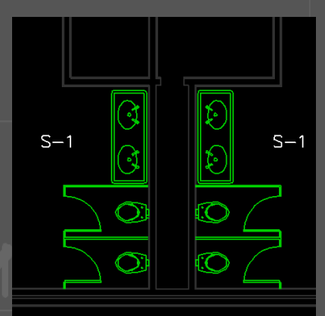
Projects should be archived at completion of each project phase. I.E. when each of the following milestones has been completed; pre-design, design, CD's, mods OR as-builts.

When a project phase is ready for archive, I'll copy the files to CD, which is submited to TIC. Additionaly, I will add the files to our archive database, so that we can easily locate them in the future. Please do not remove files from P: and take them to TIC directly, as they will not be recorded in the archive database.

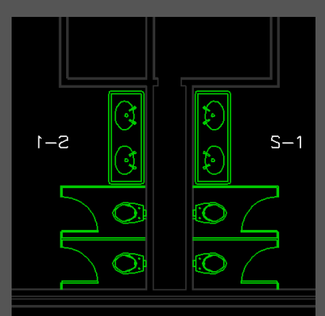
Mirror Text | ʇxǝʇ ɹoɹɹıM

There's a seldom used setting in AutoCAD which controls how text (single line and multi-line) is affected by the MIRROR command. To access this setting, type **MIRRTEXT<enter>** in AutoCAD. Usually you will want it set to 0, which retains text direction, 1 mirrors the text.

In each example below, the right side objects were mirrored to create the left side.



MIRRTEXT = 0
Original text direction is retained after mirror.



MIRRTEXT = 1
Text is mirrored. Notice the backwards 'S-1' on the left side.

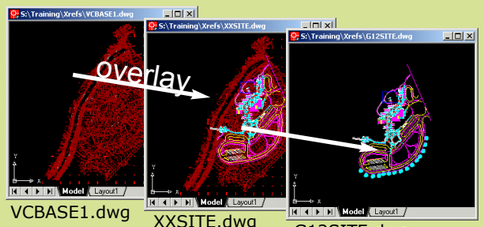
Xrefs. Attach or Overlay?

What's the Difference?

When Xreffing a drawing, AutoCAD presents you with an option to Attach or Overlay, and the choice you make only becomes important when you have nested or chained xrefs. When xrefs are attached, they work like a train. If the caboose is attached to the coal car, and the coal car is xreffed to the locomotive, then the locomotive "sees" the coal car and the caboose. Why? Because the caboose was **attached** to the coal car. Overlay's stop the train effect. In the same scenario, **overlay** the caboose to the coal car, and the caboose will not be seen in the locomotive. Here's a graphic example. Consider the following three drawing files . . .

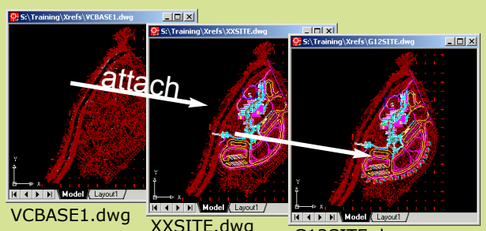
VCBASE1.dwg XXSITE.dwg G12SITE.dwg

Let's start with 3 drawing files. No xreffing at this point. From left to right; VCBASE1, XXSITE and G12SITE.



VCBASE1.dwg XXSITE.dwg G12SITE.dwg

Here we have used the **Overlay** option to xref VCBASE1 into XXSITE. Notice that in G12SITE, we **do not** see VCBASE1 (thanks to Overlay).



VCBASE1.dwg XXSITE.dwg G12SITE.dwg

Here we have used the **Attach** option to xref VCBASE1 into XXSITE. Notice that in G12SITE, we **do** see VCBASE1 (thanks to Attach).

Change Attach to Overlay or Overlay to Attach

You can change the Xref type (attach vs. overlay) once a file has been referenced. In a parent drawing (one that has an Xref), open the Xref Manager, (click Edit ➔ Xref Manager from the AutoCAD pull-down). See the Type column. To change the type, double click on the word Attach or Overlay next the the desired Xref, then OK to close the Xref Manager.

CADVENTURES

Information for Users of AutoCAD

Oct-Nov Two Thousand & One : Presented by Bruce Littlehorn for NPS and BLM

in this issue

Olé. Just say no.

Have you ever copy/pasted from Excel or Word into an AutoCAD drawing? If so, then you have used Olé. When it works, it is a great feature of Windows, but it is sometimes buggy to say the least. The following will explain how to convert your Word or Excel objects into AutoCAD objects, avoiding the problems usually associated with Olé.

A bug in PrfGenerator frequently prevents you from printing a drawing with an imbedded or linked Excel spreadsheet - which means you will not be able to print full-size to the OCE 9700.

What is Olé?

Olé (pronounced o-lay) stands for Object Linking and Embedding. Olé allows you to link (like an xref), or embed (like using AutoCAD INSERT) objects from one application into another. For example, you could link or embed an Excel spreadsheet into an AutoCAD drawing.

What's wrong with Olé?

Using OLE to link or embed spreadsheets and Word documents into AutoCAD, we have experienced problems with printing and on-screen display. A bug in PrfGenerator frequently prevents you from printing a drawing with an imbedded or linked Excel spreadsheet - which means you will not be able to print full-size to the OCE. Alternatives? Read on…

Okay, how do I avoid Olé?

Use Paste Special. Paste Special places the content of the Windows Clipboard into the current drawing, as Paste does. However, using the options we will discuss below, Paste Special creates AutoCAD entities from the graphics in the clipboard.

To Use Paste Special

1. First, we need some objects in the clipboard. Let's start with an Excel spreadsheet. Using Microsoft Excel, open **S:\Training\Paste Special\Paste Special Example.xls**. (Quick way to do this is to use Windows Explorer, navigate to S:\Training\Paste Special\, and then double-click on the file 'Paste Special Example.xls'.

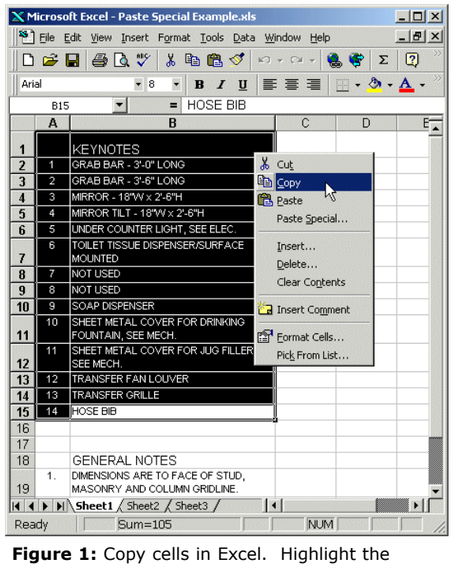


Figure 1: Copy cells in Excel. Highlight the desired cells, right-click then choose Copy.

2. Using your mouse, **highlight** the Keynotes area, then **right-click** and choose **Copy**. (Highlight an area in Excel by dragging your mouse while pressing and holding your left mouse button.

See figure 1.

The contents of the Keynotes cells are now in the Windows clipboard, ready to be inserted into your AutoCAD drawing.

3. In AutoCAD, click from the pull-down menu, **Edit → Paste Special**. Leave the selection on Paste, NOT Paste Link. Select **%PRODUCT Entities**, then click **OK**.

4. Specify the insertion point by clicking a point in your AutoCAD drawing, similar to inserting a block.

5. The objects from Excel come into AutoCAD at a scale based on your zoom factor. If necessary, scale the newly inserted objects using the SCALE command.

Notice that the objects from Excel are now AutoCAD objects; lines and text.

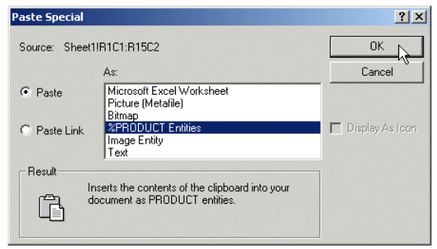


Figure 2: Paste as '**%PRODUCT Entities'** to convert the objects into AutoCAD entities.

Take Note

If you want border lines between your cells, turn them on in Excel. In the example spreadsheet used above, notice the Keynotes area has border lines, the General Notes area does not.

Be aware of cell formatting options in Excel. Notice in the General Notes area of the example Excel spreadsheet, the numbered column has cells formatted as text, rather than number. This is because I wanted a period after each number i.e. 1. 2. 3. etc. Notice also the formatting of the text column in the General Notes area; in the alignment tab, Wrap Text is enabled. Excel cell formatting options effect how the objects are imported into AutoCAD. To check or change cell formatting in Excel, highlight one or more cells, right-click and choose Format Cells.

Same General Notes in Many Drawings?

You have a series of drawings in a project, and they all require the same general notes. You could use the Olé Paste/Link feature to link one spreadsheet into multiple drawing files - but we're looking at alternatives to using Olé.

Instead of linking using Olé, begin a new/empty AutoCAD drawing, and use Paste Special as AutoCAD entities as previously described. Save this file as you would any Xref base sheet (for this example, we'll call the file X-ArchGenNotes.dwg. Then, using the XREF command attach X-ArchGenNotes.dwg into as many subsheets are you need. If you change the contents of X-ArchGenNotes.dwg, you'll see the changes in all subsheets xreferencing it.

Change Attributes

Change an Attribute's Width

Attributes are special text-like objects used in blocks. Our NPS title block/border uses attributes inside of the title block. Sometimes it is necessary to change the color or width of an attribute, and you can do both without exploding or bursting the block.

Figure 3 shows a section cut block whose attributes don't fit within the circle. You can easily change the width of attributes within a block. Use the CHTEXT command, which is included with your NPS AutoCAD tools. Let's try it . . .

1. Start with a drawing that has a block with attributes, or create a section cut like shown in figure 3 using the SECT command.

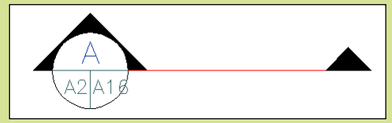


Figure 3: Before changing the attribute's width

2. At the AutoCAD command prompt, type **CHTEXT<enter>**.

3. At the Select Object prompt, select one of the attributes. Because this is a block, all attributes and the two lines will highlight. Press **<enter>** again to complete the Select Objects prompt.

4. Because we're not changing the height of the attributes, press **<enter>** to accept the default Width option.

5. Type **.75<enter>**.

6. Press **<enter>** to regen now. See figure 4.

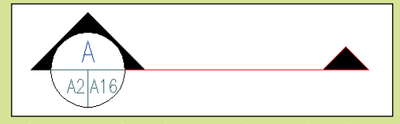


Figure 4: After changing the attribute's width to 0.75 ratio

Change an Attribute's Color

To change the color of an attribute, we will use the AutoCAD -ATTEDIT command. Notice the hyphen (-) before ATTEDIT. That's the key to accessing the command line version of ATTEDIT - required to change an attribute's color.

1. Start with a drawing that has a block with attributes, use the NPS title block, or use the drawing from the example above.

2. In AutoCAD, type **-ATTEDIT<enter>**. Be sure to include the hyphen (-).

3. At the next 4 prompts, take the default be pressing **<enter>** 4 times. . .

```
Edit attributes one at a time? [Yes/No] <Y>: <enter>
Enter block name specification <*>: <enter>
Enter attribute tag specification <*>: <enter>
Enter attribute value specification <*>: <enter>
```

4. Select an attribute, then press **<enter>** to complete the Select Objects prompt.

5. Type **C <enter>** to change the color.

6. Enter a new color.

7. Press **<enter>** to complete the command.